ADVANCED DES SIMULATIONS OF OXY-GAS BURNER LOCATED INTO MODEL OF REAL MELTING CHAMBER

Ing. Vojtech Betak Ph.D.
Aerospace Research and Test Establishment
Department of Engines
Prague, Czech Republic

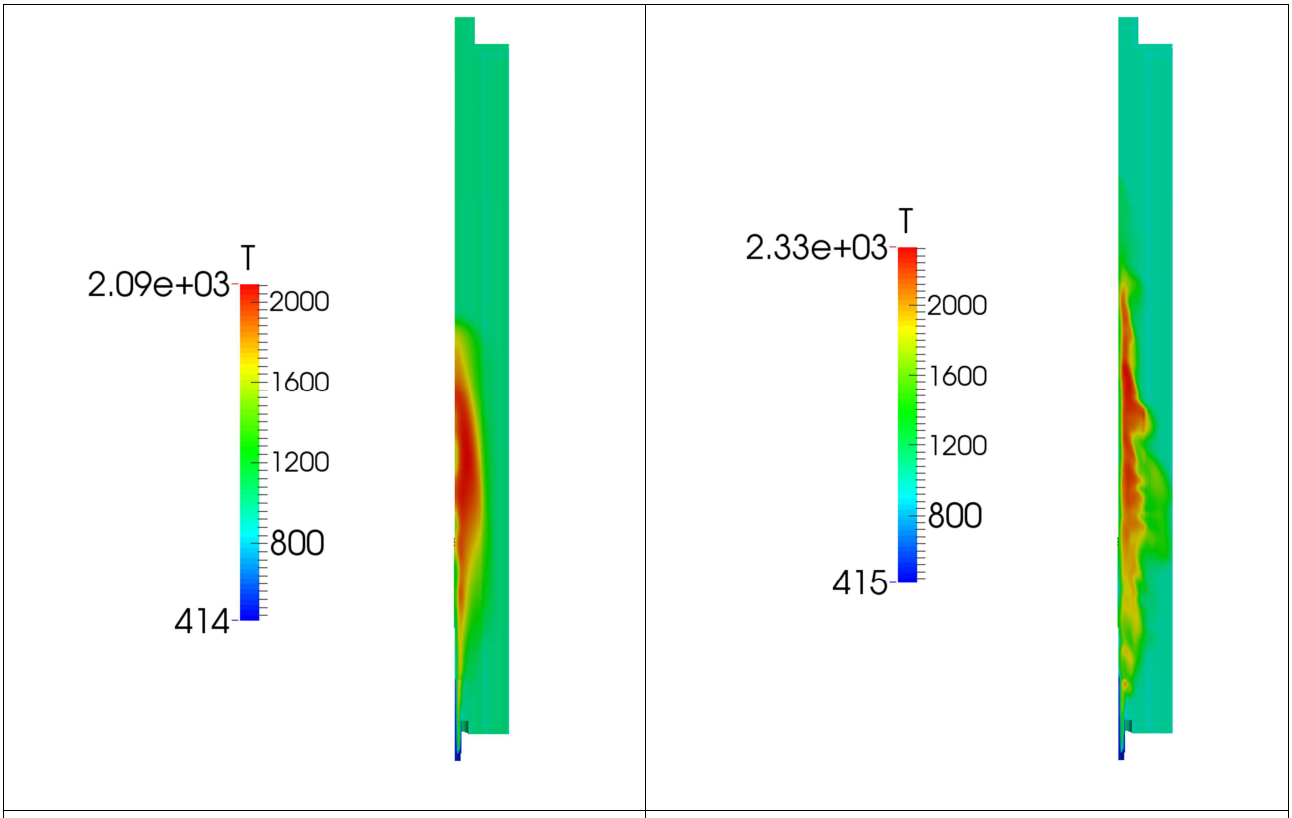
Abstract

The simulation of burner located in free stream has one disadvantage because it don't take into account a recirculation of flue gasses in real melting chamber. Combustion processes and production of emissions may be affected by this simplification. Extension of previous simulation to the model containing simplified model of test melting chamber is described in this paper. For this simulation is used an OpenFOAM platform and DES model of turbulent flow field which allow to capture unsteady phenomena of reacting flow field in free stream and to simplify modeling of turbulent flow field in near wall region. There are discussed possible improvements of solvers due very high consumption of computational time.

INTRODUCTION

The increase of computing power in last years allows to use this power into various number of simulations that allow us to better understand various phenomena. One of this phenomena is combustion. Experimental tools provide us limited amount of information due high temperature, flame radiation and highly unsteady flow. In previous work [1] was shown an advantage of Large Eddies Simulation (LES) over Reynolds Averaged Navier Stokes (RANS) turbulence mondel on isolated burner. In figure 1 is shown comparison between RANS and LES turbulence model. In the case of LES model we achieve much more information about structure of velocity and temperature field which is important for burner optimization. There are high computational demands of this model. Therefore is often used RANS model of turbulence which allow us decrease computational demands and achieve suitable results especially in the case of global parameters.

In this paper will be presented results of burner simulation which is located in model of furnace chamber. This simulation clarifies the results presented in the previous article [1]. Application of industrial burners is characterized by two timescales. The first one (shorter) is connected with mixing of fuel, it's burning and pollutant formation [1]. The second one (larger) is connected e.g. with recirculation of flue gas in furnace or heat transfer into wall or melt and is included into this simulation. There is number of simplification which will be discussed later.



a) RANS | *b) DES (LES)*

Fig. 1.: *Comparison of temperature field for different turbulence model*

GOVERNING EQUATIONS

Following system of filtered Partial Differential Equations (PDE) is used for description of reacting flow field

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0 \tag{1}$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}[(\mu + \mu_t)(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij})] \tag{2}$$

$$\frac{\partial \rho h}{\partial t} + \frac{\partial}{\partial x_j}(h\rho u_j) = \frac{Dp}{Dt} + \frac{\partial}{\partial x_j}[(\alpha + \frac{\mu_t}{Pr_t})\frac{\partial h}{\partial x_j}] + S_h \tag{3}$$

$$\frac{\partial \rho Y_i}{\partial x_j} + \frac{\partial}{\partial x_j}(\rho Y_i u_j) = \frac{\partial}{\partial x_j}[(\mu + \mu_t)\frac{\partial Y_i}{\partial x_j}] + S_Y \tag{4}$$

where ρ is a density, u_i is a component of velocity vector, p is a pressure, h is an enthalpy, Y_i is a mass fraction of species i, μ is a dynamic viscosity and α is a heat diffusion coefficient. S_Y and S_h are source terms in relevant equations. μ_t represent a turbulent or subgrid-scale viscosity and Pr_t is turbulent Prandtl number.

These equations have to be completed by equation of state $p = f(\rho)$. First we have to define molar fraction as

$$y_i = Y_i \frac{M}{M_i}. \quad (5)$$

Where M is an average molar mass and M_i is a molar Mass of fraction i. Then we apply Dalton Law

$$p_i = y_i p \quad (6)$$

where $p_i$is a partial pressure of fraction i. Now it is possible to define ideal gas law for fraction i

$$p_i = \rho_i n_i R T \quad (7)$$

In this equation we used partial density ρ_i and number of moles n_i of fraction i, temperature T and universal gas constant R. Finally we obtain density in following form

$$\rho = \sum y_i \rho_i \quad (8)$$

For ideal gas the enthalpy is defined as

$$h = c_p T \quad (9)$$

For ideal mixture is coefficient of heat capacity c_p defined as

$$c_p = \sum y_i c_{p_i} \quad (10)$$

where c_{p_i} is a heat capacity of fraction i.

Turbulent viscosity is then modeled via suitable model [2,3]. In this work the Detached Eddy Simulation (DES) is used. This model is a combination of RANS and LES models. In this model the near-wall regions are solved as RANS equations and the other flow as LES. It leads to reduction of computational cost. In this case was use k-omega SST DES turbulence model presented in [4].

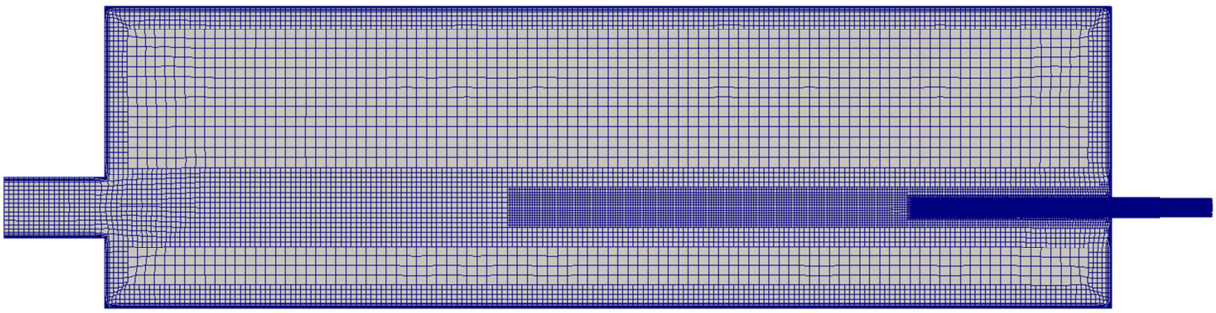
Combustion of natural gas is simulated Partial Stirred Reactor model [5] and 4-step chemistry model [6] with one step chemistry for NO_x [7].

Tab. 1.: *Reduced chemistry model used in simulations*

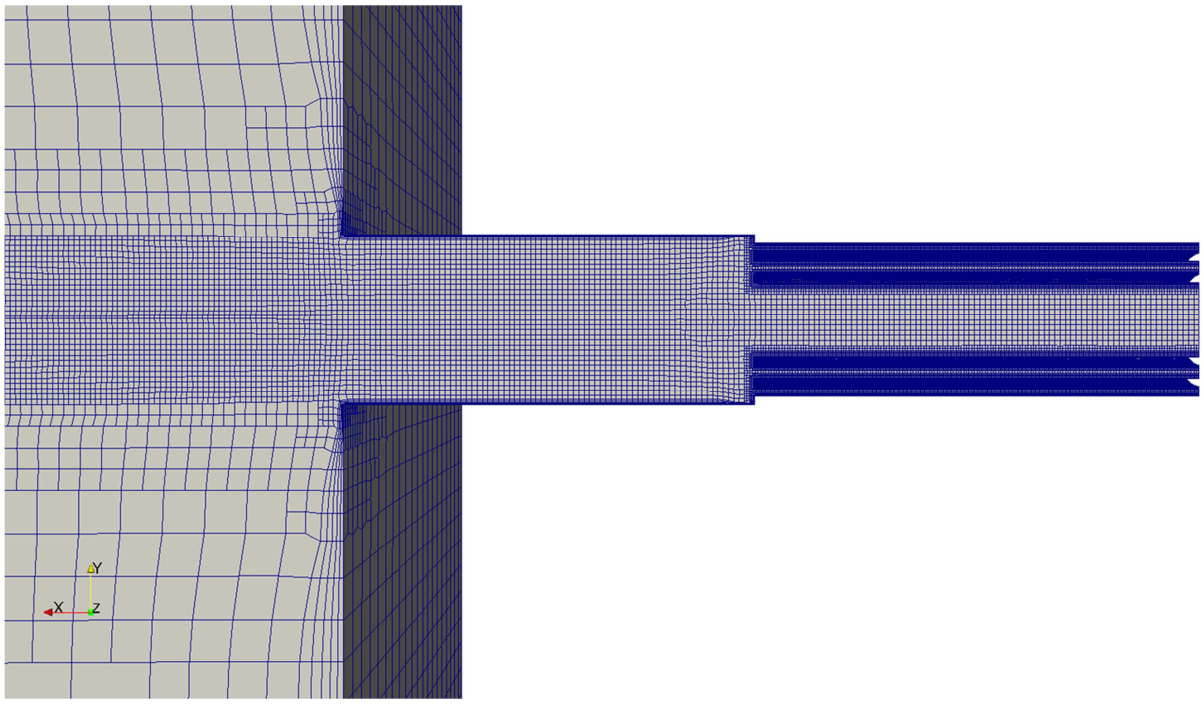
	A	n	E_a
$CH_4^{0.5} + 0.5O_2^{1.25} => CO + 2H_2$	7.824E+13	0.0	30000
$CH_4^{1} + H2O^{1} => CO + 3H_2$	3E+11	0.0	30000
$H_2^{0.5} + 0.5O_2^{1.5} <=> H_2O^{1}$	1.209E+18	0.0	40000
$CO^{1} + H_2O^{1} <=> CO_2^{1} + H_2^{1}$	2.75E+12	0.0	20000
$N_2^{1} + O_2^{0.5} <=> 2NO$	7.4E+13	-0.5	76337.13

RESULTS

Geometry of computational domain is shown in figure 2. It consists from 2.32 M of cells. SnappyHex meshing algorithm is applied in order to create hexahedral dominant mesh. Boudnary layer is resolved via wall functions which allow height of first layer $y^+ > 30$.



a) global overview

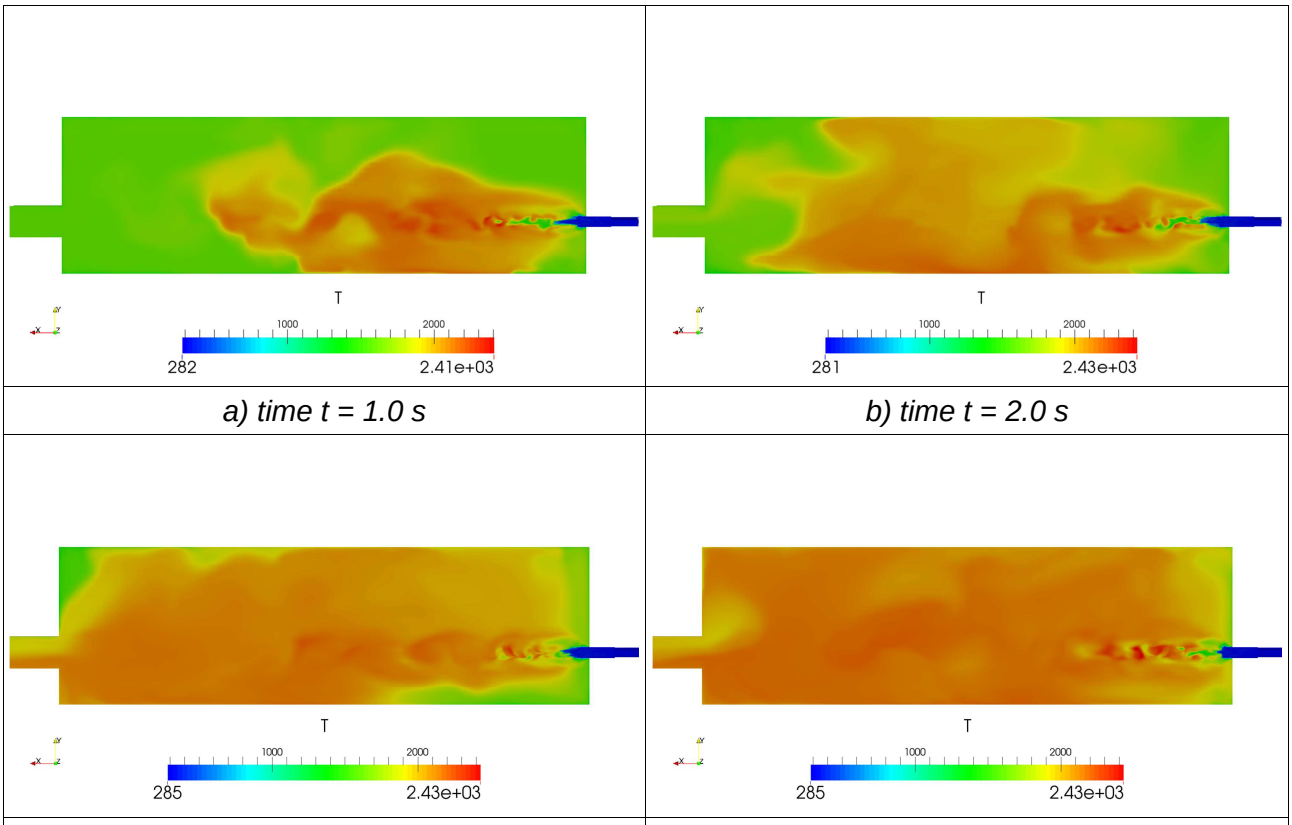


b) burner detail

Fig. 2.: *Computational domain*

In this case was solved problem where the ratio between primary and secondary oxygen stream is 40/60 and air fuel ratio (AFR) is 4. This value corresponds to stoichiometric ratio between fuel (natural gas) and oxygen. The ambient temperature of furnace gases is set to 1723 K and it is composed from 43 % of water steam, 47 % of carbon dioxide and 10 % of nitrogen.

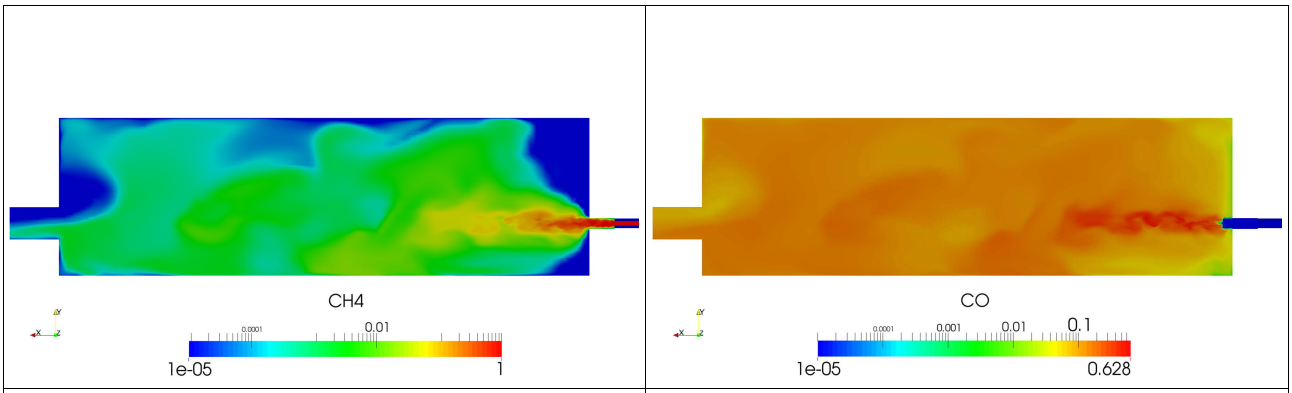
In figure 3 is shown time evaluation of temperature field on symmetry field. It is possible to seen that we need to solve more than 4 s of physical time to obtain more realistic temperature field. Therefore we need is much more time to solve this problem than we need for simulation of isolated burner.



a) time t = 1.0 s | *b) time t = 2.0 s*

a) time t = 3.0 s | *b) time t = 4.0 s*

Fig. 3.: *Evaluation of temperature field*



a) concentration of CH_4 | *b) concentration of CO*

Fig. 4.: *Concentration of CH_4 and CO on symmetry plane in time t=4,0 s*

Figure 4 shown emissions in combustion chamber. We obtain similar results as in previous work [1] where simplified model predict unburned CH_4 and CO on outlet from combustion chamber. This could be improved by setting different ratio between primary and secondary oxidizer stream ratio or by changing geometry of furnace chamber which allow longer combustion process.

CONCLUSIONS

The result of the burner simulation which is located in the model chamber is summarized in this paper. We obtain similar results to our previous work presented in [1]. This simulation is much more time-consuming because we need to resolve two different time scales. The short one is connected with mixing fuel and oxidizer and burning. The large one is connected with recirculation of flue gases in furnace chamber.

Several simplification have been applied due limitations connected with implementation of OpenFOAM platform. The main one is connected with modeling heat losses and radiation. Explicit form of boundary condition for descriptions of heat sources/loses is implemented in OpenFOAM. This form of boundary condition is for high thermal fluxes (>1000 W/m^2) unstable. This numerical instability leads into reduction of time step and therefore is there significant increase of simulation time.

There are two possible themes for future work. The first one is implementation of implicit form of heat loses boundary condition which leads to higher stability of simulation. The second one is implementation of solvers which allow to solve parallel combustion (highly unsteady region) and recirculation of flue gasses (steady region) in combustion chamber.

REFERENCES

[1] Betak, V., Kubata, J.: Large Eddies Simulation of FlammaTec Oxy-Gas Burner, 13th International seminar on Furnace Design, Velke Karlovice, 2015

[2] cfd-online.com: *List of RANS model*, http://www.cfd-online.com/Wiki/RANS-based_turbulence_models

[3] cfd-online.com: *Large Eddy Simulation, l*http://www.cfd-online.com/Wiki/Large_eddy_simulation_%28LES%29

[4] F. R. Menter, F. Kuntz, R. Langhtry: *Ten Years of Industrial Experience with the SST Turbulence Model*, Turbulence, Heat and Mass Transfer, 2003

[5] Golovichev, V.: *Turbulent combustion modeling*, http://www.tfd.chalmers.se/~valeri/book-chapt7.pdf

[6] Bibrzycki, J., Poinsot, T.: *Reduced chemical kinetic mechanism for methane combustion in O_2/N_2 and O_2/CO_2 atmosphere*

[7] Fedina, E., Fureby, C.: *Combustion LES of CESAR Multi-Burner Annular Combustor*, 49th AIAA Aerospace Sciences Meeting, 2011